

COMPUTATIONAL ANALYSIS OF FLOW IN 3D PROPULSIVE  
TRANSITION DUCTS

by

Paavo Sepri  
Associate Professor  
Mechanical and Aerospace Engineering Department  
Florida Institute of Technology  
Melbourne, FL 32901

The main focus of this investigation has been to undertake a numerical analysis of fully three dimensional, statistically steady flows in propulsive transition ducts that are being considered for use in future aircraft of higher maneuverability. The purpose of the transition duct is to convert axisymmetric flow from conventional propulsion systems to that of a rectangular geometry of high aspect ratio, so that downstream thrust deflectors may be applied to effect pitch and yaw moments. In an optimal design, the transition duct would be of minimal length in order to reduce the weight penalty, while the geometrical change would be gradual enough to avoid detrimental flow perturbations. Recent experiments conducted at the Propulsion Aerodynamics Branch [1] have indicated that thrust losses in ducts of superelliptic cross-section can be surprisingly low, even if flow separation occurs near the divergent walls. However, from a material standpoint, it may be unacceptable to accommodate the greatly increased heating which accompanies flow disturbed by separation, secondary vortices, or increased turbulence.

The present investigation has proceeded along three avenues, with a fourth component involving turbulence modeling being considered for future continuation. In order to address the objective of developing a rational design procedure for optimal transition ducts, it is necessary to have available a reliable computational tool for the analysis of flows achieved in a sequence of configurations. Current CFD efforts involving complicated geometries usually must contend with two separate but interactive aspects; namely, grid generation and flow solution. The first two avenues of the present investigation have been comprised of suitable grid generation for a class of transition ducts of superelliptic cross-section, and the subsequent application of the flow solver PAB3D to this geometry. The code, PAB3D, has been developed at the Propulsion Aerodynamics Branch as a comprehensive tool for the solution of both internal and external high speed flows. A document for its operation is currently being refined by its principal developer, K. S. Abdol-Hamid [2]. Helpful guidance in the aspects of grid generation is gratefully acknowledged by the author to S. P. Pao, who has recently reported results [3] of his adaptive grid methods jointly with K. S. Abdol-Hamid.

The third avenue of investigation has involved analytical formulations to aid in the understanding of the nature of duct flows, and also to provide a basis of comparison for subsequent numerical solutions. Attention is drawn to the fundamental analytical solution derivable for fully developed laminar duct flow

of rectangular cross-section. This solution explicitly clarifies the seemingly paradoxical corner flow limit, in which a vanishing velocity with vanishing gradients still maintains a non-vanishing curvature, so that viscous retardation can indeed counterbalance an axial pressure decay. It is suggested that this physical balance may be connected with the well known appearance of corner vortices, and therefore, that numerical simulations of corner flows must adequately represent the velocity Laplacian in this neighborhood. In a parallel analytical effort, a closed form solution has been derived for the incompressible 3D Navier-Stokes equations, which are driven by a particular volumetrically distributed force field. The solution is adapted to the geometry and boundary conditions of a duct of rectangular cross-section, and it characterizes flow similar to that which has been observed experimentally. This solution provides the three benefits of aiding in the physical understanding of the role of Reynolds stresses, aiding in the correct application of boundary conditions, and aiding in the assessment of the accuracy of numerical flow solvers.

Numerical results to date include the generation of two preliminary grid systems for duct flows, and the initial application of PAB3D to the corresponding geometries, which are of the class tested experimentally. The first grid system discretizes the volume bounded by superellipses of evolving exponent described in Reference 1, and which is denoted as configuration 2 therein. The second grid system corresponds to a duct comprised of evolving rectangular cross-section, with the same height and width variation as in configuration 2. The rectangular duct has been included in the investigation for three reasons. First, the simplicity of the geometry permits a quick and unambiguous application of a computational mesh, which is easily concentrated in the wall and corner neighborhoods. Second, this duct retains all the essential flow features desired for study in the more practical configurations. Third, the rectangular duct highlights the important possibility of streamwise corner vortex generation, which could influence the design of optimal ducts. Preliminary mesh generation for the superelliptical duct has also been completed to the extent that PAB3D accepts this mesh for computation without unduly ill effects. However, refinement of the superelliptical gridwork is necessary for obtaining numerically accurate results.

At the writing of this report, only the initial computations utilizing PAB3D have been completed. These have indicated the appearance of the main flow features anticipated for such duct geometries, but it has also become apparent that the preliminary outflow boundary condition utilized must be modified for this subsonic case. Incomplete specification of such boundary conditions has been a problem recognized by various investigators. For the present study, a downstream nozzle will be added to the configuration in order to provide a supersonic exit condition, which also simulates the actual case that has been tested.

Recommendations for continued investigation fall into five categories; namely, grid refinement, boundary condition evaluation, turbulence modeling and utilization, implementation of the analytical test case, and comparison of converged solutions with the available experimental results. Some of these

should be completed within the time remaining in the summer tenure, while research support will be sought for further continuation also.

### References

1. Burley, J. R. II; Bangert L. S.; and Carlson, J. R.: "Static Investigation of Circular-to-Rectangular Transition Ducts for High-Aspect-Ratio Nonaxisymmetric Nozzles," NASA Technical Paper 2534, March 1986.
2. Abdol-Hamid, K. S.: "A Multiblock/Multizone Code (PAB3D-v2) for the Three-Dimensional Navier Stokes Equations: Preliminary Applications," NASA CR-182032, 1990.
3. Pao, S. P.; and Abdol-Hamid, K. S.: "Application of a New Adaptive Grid for Aerodynamic Analysis of Shock Containing Single Jets," AIAA Paper 90-2025, AIAA/SAE/ASME/ASEE 26th Joint Propulsion Conference, July 16-18, 1990.